International Journal of Mechanical and Production
Engineering Research and Development (IJMPERD)
ISSN (P): 2249-6890; ISSN (E): 2249-8001
Vol. 7, Issue 4, Aug 2017, 381-388

DESIGN AND ANALYSIS OF TWO THROAT WIND TUNNEL

SHIVA PRASAD .U[1], B. RAMAMOHAN PAI[2], YOGEESHA PAI[3],
D. GOVARDHAN[4] & B. PRAVEEN[5]

[1]*Assistant Professor, Department of Aeronautical Engineering, IARE, Dundigal, Hyderabad, India*
[2]*Professor, Department of Aeronautical & Automobile Engineering, MIT-MU, Manipal, India*
[3]*Assistantprofessor, Department of Aeronautical & Automobile Engineering, MIT-MU, Manipal, India*
[4]*Professor, Department of Aeronautical Engineering, IARE, Dundigal, Hyderabad, India*
[5]*Assistant Professor, Department of Aeronautical Engineering, MLR Institute of Technology, Hyderabad, India*

ABSTRACT

To aid in understanding the concepts of aerodynamics, wind tunnels are used. The aim of this paper is to design and analyze a two throat supersonic wind tunnel to give a better start-up condition at supersonic speeds. As the tunnel design involves high complexity, it is generally convenient to divide into segments. Initially, nozzle segment is developed using the method of characteristics (MOC) because, a careful designed nozzle is vital for tunnel function. In the next step, a settling chamber is introduced at the nozzle exit to straighten the flow, just before it enters the test section and the diffuser is designed to reduce the losses related to shocks attached. As we know that second throat at the diffuser section plays an important role in starting of a supersonic tunnel, so the area of the second throat is increased. In next phase, all segments are integrated and analyzed using the commercial code. Numerical analysis carried out reveals that the acceptable numerical accuracy is achieved with practice levels of grid resolution (300 x 30,) with the assumption that in absence of separation, low turbulence intensifies due to the effect of large acceleration along the downstream of the tunnel. The flow solutions give a better insight of understanding the flows, to investigate the effect of numerical code and grid independence. From this work, the necessity of grid dependence study and to the quantification of numerical error has been addressed by representing the computational results in complicated airways.

KEYWORDS: CFD, Design, Grid, MOC, Mach Number, Start-Up & Wind Tunnel

Received: Jul 04, 2017; **Accepted:** Jul 28, 2017; **Published:** Aug 07, 2017; **Paper Id.:** IJMPERDAUG201738

Original Article

INTRODUCTION

A wind tunnel is a device designed to generate free stream flows at various speed regimes in the test section. The design feature of the test section is to streamline the flow, getting into the test section of a wind tunnel, in order to record the valid test information. This objective becomes tougher to attain because, the ratio of the flow will increase from the subsonic regime to the supersonic regime wherever shock waves could form. For which, the divergent portion will be given more importance for the supersonic nozzle contour, in particular, the straightening section is extraordinarily necessary for this reason. The contour of enlargement primarily depends on the sonic line. This has already proved in [8] that results obtained using the method of characteristics with the assumption of a sonic line was well matched to experimental results. Wind tunnels are used to investigate the flow over a solid surface at various conditions. Using this, aerodynamic data from models permit engineers to work on inexpensively weak designs without building the highly inexpensive and fully-functional prototypes. The study of aerodynamics is becoming increasingly important in the understanding of the forces object experiences, as it moves through air.

Over the recent years, although abundant advances have made in theoretical and computational methods, wind tunnel tests are still essential in many practical problems to finalize the design decisions. In today's world, although growing computational powers are advancing towards new technologies, but the use of wind tunnels to resolve aerodynamic glitches may seem obsolete.

DESIGN METHODOLOGY

Nozzle

In this paper, the tunnel test section contour is designed using the method of characteristics, which results shock free flows. The method characteristics are a technique used to find a solution for the velocity potential to a particular boundary condition. Velocity potential influences the partial differential equations into a system of differential equations. There is a unique combination of curves known as characteristic curves, for which the latter one holds true. The MOC equations will be solved along the flow lines which have formed into flow net, these equations are solved for pressure waves propagating on the Mach lines in the direction, parallel to the speed vector of the flow [5]. As the presence of shock waves in viscous flow results in a huge pressure gradient, boundary layer separation due to the interaction between shock waves and boundary layer [4]. So the Prandtl-Meyer flow will seek to avoid the formation of oblique shock wave in the order to straighten the flow.

Numerical solution involving a finite number of grid points is susceptible with a truncation error, when the higher order terms are neglected. Due to Rounding of a number to a significant figure in the flow calculation by digital computers, the results are subject to the round-off error [6]. Accurate results can be obtained by implementing first and second precautions in the calculations and the latter one, while performing the experiment respectively as mentioned below.

- Adverse pressure gradients can be avoided.
- Boundary layer thickness should be carefully computed by having an adequate number of cells on the wall streamline.
- Through suction large initial boundary should be discarded.

Settling Chamber

The critical component for the wind tunnel that is settling chamber is focused on this section, which is located before the test section as shown in figure 1. The settling chamber consents the flow to straighten. Turbulent flows in the settling chambers may result in nutritional forces in the test section. Reduction in turbulence leads to a better wind tunnel, which will simulate actual flying conditions.

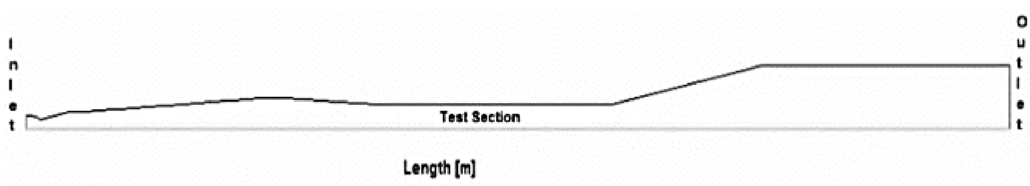

Figure 1: Geometry of Wind Tunnel

Test Section

In wind tunnel, the test section is the significant part, which should be designed according to the flow requirement

with high priority, based upon the specific need and the flow velocity. According to test section requirement, the other sections can be modeled accordingly.

Diffuser Design

The most fundamental design parameter to be followed in the design of the wind tunnel is the ratio of diffuser throat area to nozzle throat area. The flow inside the supersonic diffuser is complicated by shock-shock interaction, shock-boundary interaction, etc., with a resultant loss in stagnation pressure [2] [9]. The diffuser function is to reduce the speed with little loss in total pressure. The deflection angle of wall is set as 11.30^o for the convergent section; it is selected from the θ-β-M relations and the area ratio of nozzle to diffuser is maintained as 0.70. For any upstream Mach number M, there is a maximum deflection angle θ_{max}. There exists no solution for oblique shock wave, if the physical geometry angle θ is greater than $\theta_{max.}$ Instead, nature establishes a curved shock wave, detached from the corner. In the present case, as the diffuser deflection angle is greater than θ_{max}, Lamda (λ) shock wave has formed (i.e., due to the influence of the boundary at walls the normal shock has changed to oblique at the walls given a combination of normal and oblique shock) at the entry of the diffuser section. In order to obtain the required mass flow rate through the nozzle, the throat area should be small when compared to the diffuser area [3]. The main objective of the design is to account the flow blockage and stagnation pressure loss by swallowing the starting normal shock, to occur this size of the diffuser throat should be suitably large.

NUMERICAL DISCRETIZATION

The computational technique deals with the calculation of the flow field properties of discrete points in the flow, resulting from a Taylor series expansion of these flow properties. For specific boundary conditions, one constructs, in an exceedingly stepwise fashion, a "characteristics, net" of no matter abstraction resolution one would really like. One will begin with a rough grid and acquire solutions with in turn finer nets, until 2 ordered solutions comply with a desired decimal.

Except for very modest cases, The PDEs which govern fluid flow are not generally acquiescent to analytical solutions. Therefore, in order to analyze the computational results, the fluid domain needs to discretized into smaller sub domains. Control of the refinement and/or coarsening via the error indicators is often undertaken by using either the 'solution gradient' or 'solution curvature'. Hence, in order to account this, grid adoption need to contemplate on all its limits while solving this for variable coupled refinement method [6]. For analyzing the results, mesh tools and solvers prefer to join exposed elements together in what is known as a face mesh, which is preferred to join the exposed elements together for the purpose of applying boundary conditions and for rendering the mesh domains. Grid adoption is often referred to modify the existing mesh so as to accurately capture the flow physics. The objective these modifications are to capture the physics of flow without increase of additional computational effort. The final design of wind tunnel was then discretized with different levels of mesh was adopted to check for grid dependency.

FLOW SIMULATION USING FLUENT

ANSYS Fluent has broad capabilities of solving the fluid flow problems to meet the user requirements and to analyze the flow physics. The equations that have been derived within the preceding sections apply to a viscous flow. The additional transport development of mass diffusion has not been enclosed as a result is restricted to a regular, non-chemical reacting gas.

Turbulence Model

The two equation turbulence model k-ε is one of the most common turbulence models, although it just doesn't perform well in cases of large adverse pressure gradients. This model includes two extra transport equations to represent the turbulent properties of the flow, which allows accounting for history effects like convection and diffusion of turbulent energy.

Transport equations for standard k-epsilon model.

For turbulent kinetic energy k

$$\frac{\partial \rho}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_i}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right]+P_k+P_b-\rho\varepsilon-Y_M+S_k$$

For dissipation ε

$$\frac{\partial}{\partial t}(\rho\varepsilon)+\frac{\partial}{\partial x_i}(\rho\varepsilon u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right]+C_{1\varepsilon}\frac{\varepsilon}{k}\left(P_k+C_{3\varepsilon}P_b\right)-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}+S_\varepsilon$$

Flow Physics

In computational simulation, the boundary conditions play a major role to match with the domain with actual physics. The solution is dependent on the value we set and the result may vary accordingly. Incorrect sets of boundary conditions may introduce impact of the solution, while a proper set of boundary conditions can avoid that. So, setting boundary conditions for a variety of problems is very noteworthy. At the same time, different problems in the environment have different boundary conditions according to their physical requirements. The error is the difference between the actual solution and the obtained solution, which is obtained from a differential operator to difference operator, which is called the discretization or truncation error. The implicit second order method was used. This method involves the derivatives of the next time level. Due to this reason, they are iterative in nature and the discretization error was set at 10^{-4} for better accuracy. Initially, the pressure was set to 14.2 with temperature 775.14k to initialize the flow.

Table 1: Mesh Statistics

Domain	Level	Mesh Size	Nodes	Elements
Fluid	A	200 x 20	8442	4000
	B	250 x 25	13052	6250
	C	300 x 30	18662	9000
	D	400 x 40	33768	16000

RESULTS AND DISCUSSIONS

Once the solution is converged with the desired level of accuracy, the results were analyzed for grid dependency.

Grid Dependency

As we all know that the numerical results will be sensitive to the resolution of the mesh. In order to reduce the sensitivity impact on the solution, the grid needs to be refined, so that the numerical solution of the Navier–Stokes equations becomes more and more accurate. The reason behind this is the effect differential equations in the way they are treated by the difference quotients. This variation is addressed in the way they are expressed, using the values of the flow

quantities, that is by using different computational grid and the statistics which is presented in table 1. Hence, more the refinement in the grid the more accurate the differentials can be approximated and the better spatial gradients of flow magnitudes are determined [6]. To obtain the accurate solution, the grid resolution should be high to achieve better results and to resolve the physical scales of the problem [1]. However, the concentrated grid resolution that is practically realistic is limited by the available computational power and time. For this reason, relatively coarser grids are used.

From figure 2, the variation of mach number with respect to the axial length for different levels of grid, it is seen that there is minutute deviation observed in the range of 0m - 5m & 15m- 20m, which are the inlet and diffuser sections of the wind tunnel. In which, maximum variation in the curve is observed at the exit of the diffuser, with the level A (200x20) plot showing the subsonic mach number at the exit (0.7M) and the level B (250x25) showing maximum exit Mach number among the 4 different levels, which is due to the coarseness of the mesh as discussed earlier. Almost all grid levels show that the maxinmun mach number of the test section reaches to 2.75M. There is a very small deviation, for different levels of the mesh grid dependency test suggest that the results are in the with acceptable accuracy, with the least error.

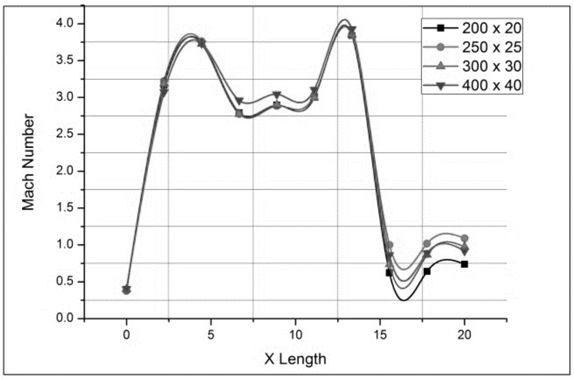

Figure 2: Mach Number vs Distance(m)

In figure 3, contour of pressure, we observe that there is a very high pressure loss across the diffuser, which shows reduced diffuser efficiency. This is because of the diffuser design having only one divergent section to reduce the Mach number of the flow, instead of the common convergent divergent diffuser type.

From the figure 4 & figure 5, dynamic pressure plots suggest that the grid levels A, B & C have very small variation and level D mesh is slightly varied compared to the overall levels, suggesting that grid dependency is within the limits and with desired level of accuracy.

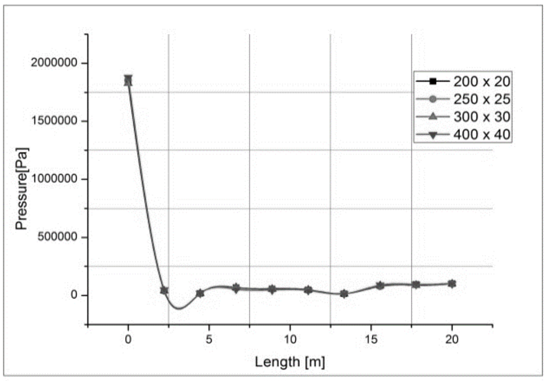

Figure 3: Pressure vs Distance

Plot of Dynamic Pressure vs Distance

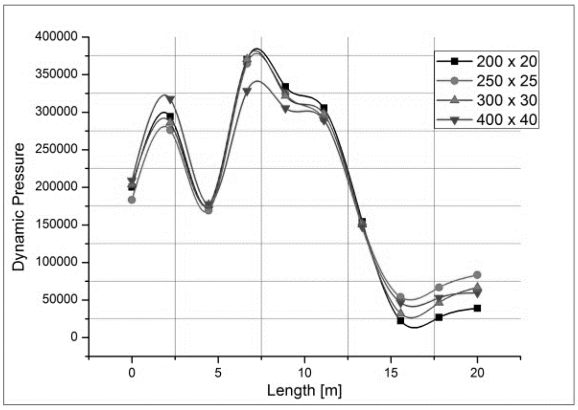

Figure 4: Dynamics Pressure vs Distance

Plot of P/P_0 vs Distance

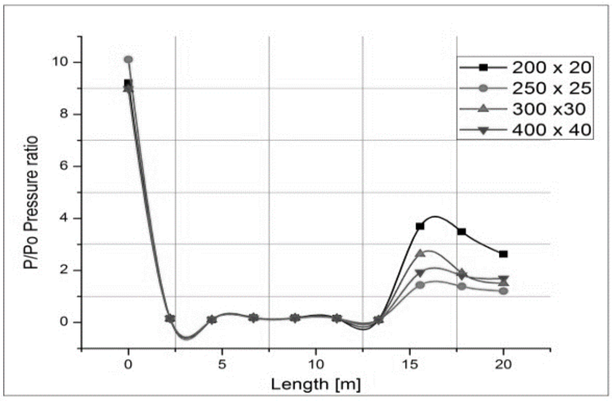

Figure 5: Relative Pressure Ratio vs Distance

Validation of Results

Validation of the above plots is done by comparison with different levels of the grid. The aforementioned plot of figure 5 still largely show similarities with the relative pressure, dropping down in the test section and then showing a gradual increase from the diffuser section onwards, which is the desired criteria for the supersonic wind tunnel design, and to obtain the desired flow mach number in the tunnel test section which is base for validation of the problem data. As compared to the grid levels, there is a slight variation only of the diffuser section which is due to increase in turbulence and shocks formed.

The plot of figure 6 of Coefficient of pressure in the axial direction shows continuous uniformity without any deviations among the different levels of mesh.

Plot of Coefficient of Pressure vs Distance

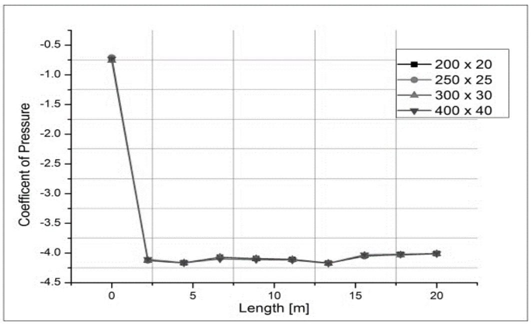

Figure 6: Coefficient of Pressure vs Distance

Comparison of the λ Shock Formation at the Entry of the Diffuser Throat for Different Levels of Mesh

Figure 7: Mach Contour of Mesh 200 X 20

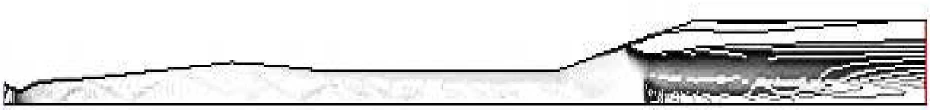

Figure 8: Mach Contour of Mesh 250 X 25

Figure 9: Mach Contour of Mesh 300 X 30

Figure 10: Mach Contour of Mesh 400 X 40

From the figure 7 to figure 10, it is observed that for different levels of mesh the formation of shock is observed at the diffuser section, which turns the flow Mach number to subsonic levels, giving rise to lamda shock for levels A to C, the shock is clear in level D, the contour plot is slightly varied but as compared to the flow properties, the variation is in reasonable limits. The flow in the tunnel section is smooth with very slight introduction of turbulence, which is due to the boundary layer thickness.

Start and Unstart Conditions

The phenomenon which occurs, when an upstream mass flow is greater than the downstream mass flow is called the supersonic choking, which is an unstart condition in the nozzle flows. This phenomenon results in mismatch of mass flow which cannot propagate downstream in contrast to subsonic flow. In case of supersonic flow, the misalliance will be

carried behind a terminal shock wave resulting in subsonic gas velocity. In order to reach equilibrium, the normal shock wave then propagates upstream as an effective acoustic velocity. Even in other ways, unstart can be conceptualized, that is alternatively this can be treated by decreasing the stagnation pressure inside the supersonic duct; by which the upstream stagnation pressure can be increased than the downstream stagnation pressure. This can result even by decreasing the throat size in supersonic ducts [8]. This is possible, even if the diffuser throat is less than entrance throat. This change will result in decreasing mass flow, which leads to the condition of unstart. The inlet was set to an initial pressure of 15 atm (gauge pressure = 9 atm), for which the unstart condition formed. Stable operating conditions were established for an initial pressure of 20 atm (gauge pressure =14.2 atm).

CONCLUSIONS

A supersonic wind tunnel with capability of testing up to 2.75 Mach was designed and analyzed. The method of characteristics proved to be an ideal procedure for the design of the supersonic nozzle free of shocks. The optimum area ratio of the nozzle throat to the diffuser throat was found to be 0.71. If the diffuser throat is not large enough to allow the startup normal shock to pass through the tunnel, it unstarts. A conventional subsonic diffuser was designed which resulted in flow separation by interaction with the shock wave(s) and reduced the losses related to shock attached at the diffuser throat section. Grid dependency test was carried out, and the results were analyzed for different levels of mesh and found to coincide, thus indicating stable design. There was very minute turbulence occurring in the test section, indicating that the flow in the test section was not smooth and streamlined as desired for testing, which is due to the fact boundary wall roughness (which can be neglected in the present analysis as the turbulence level is very less). Design conditions were analyzed for various geometries and operating speeds, which showed the start and unstart conditions.

REFERENCES

1. *Bholanath and K. Srinivasan, (2007). Design Methods for Axisymmetric Supersonic Nozzle ContoursInternational Journal of Turbo and Jet Engines, 24, p. 115–118.*
2. *Hyungrok do, Seong-KyunIm, M. Godfrey Mungal, mark A. Cappelli, 2011. Supersonic inlet duct unstart induced by fuel jet injection, mechanical engineering department, Stanford University, Stanford ca 94305.*
3. *James w. Hale, 1964, Influence of pertinent parameters on ejector-diffuser performance with and without ejected mass, rocket test facility, ARO, Inc. Technical documentary report no. Aedc-tdr-64-134.*
4. *B. G. Mingazov and I. S. Davletshin, 2011, On Choice of Turbulence Models and Grid Parameters for Computation of Flows in Diffuser Ducts, published in Izvestiya VUZ. AviatsionnayaTekhnika, 4, 24–28.*
5. *Shiva Prasad U. et.al, 2017, Numerical Analysis of The Effect of Nozzle Geometry on Flow Parameters in Abrasive Water Jet Machines, Pertanika Journal of Science and Technology, 25, p. 497-506.*
6. *S.Pugazenthi and Andy C.McIntosh, 2011, Design and Performance analysis of a supersonic diffuser for plasma wind tunnel, International Journal of Mechanical and Materials Engineering 2:1.*
7. *Dr. Srinivasa Rao, 2010, Modeling of Turbulent Flows and Boundary Layer, INTECH, Croatia, Source: Computational Fluid Dynamics, January, 420.*
8. *Vladimir v. Lunev, 2009, Real gas flows with high velocities, CRC press by Taylor & Francis group.*
9. *R. T. Biedron* and T. C. Adamson Jr. 1988, Unsteady Flow in a Supercritical Supersonic Diffuser, AIAA Journal 1336, 26, 11.*